2023 Annual Conference & Exposition
Baltimore Convention Center, MD | June 25 - 28, 2023
The Harbor of Engineering Education for 130 Years
Paper ID #37876

The Design of an Applied Computational Fluid Dynamics and Heat Transfer Course Facilitating the Cloud Computation Technology

Dr. Wenhai Li, Farmingdale State College

Assistant Professor in Department of Mechanical Engineering Technology, Farmingdale State College, Farmingdale, NY 11735

Dr. Foluso Ladeinde, Stony Brook University

The Design of an Applied Computational Fluid Dynamics and Heat Transfer Course Facilitating the Cloud Computation Technology

Wenhai Li
Department of Mechanical Engineering Technology
Farmingdale State College
2350 NY-110, Farmingdale, NY 11735

Foluso Ladeinde
Department of Mechanical Engineering
113 Light Engineering Building
Stony Brook University, 100 Nicolls Road,
Stony Brook, NY 11794-2300

Abstract

Traditional computational fluid dynamics and heat transfer (CFDHT) courses, which are mostly at the graduate level, focus on the application of numerical methods to solve the system of nonlinear differential equations that govern fluid flow and heat transfer, with some initial and boundary conditions. However, due to the limited computational resources available for classroom instruction, the problems used for illustration and laboratory assignments are limited to simple canonical types. This means that students will not be able to analyze realistic problems with practical applications, which are inherently complicated, computationally expensive, and require high-performance computing (HPC) clusters that take advantage of massive parallelization. In this paper, a course in the curriculum that addresses this issue is proposed. In this course, the fundamental theories of high-performance computing will first be introduced. Then, a commercial CFDHT package, AEROFLO Cloud, operated based on a Software-as-a-Service (SaaS) model, will be presented. The software is deployed on a Cloud server that has been optimized for high-performance computing and is accessible via a web browser. Students will use the software package to set up realistic CFDHT projects, run the simulations on the cloud, and visualize and post-process the simulation results on the cloud. The modeling and visualization tasks can be carried out with a personal classroom computer (PC) with an Internet connection. Several laboratory (simulation) projects based on practical applications are proposed, and the methods required to analyze the simulation results will be taught. In terms of the seven current ABET Student Outcomes (SOs), it is envisioned that, with the appropriate performance indices, the course will contribute to satisfying SO (1): Identify, formulate, and solve complex engineering problems, SO (5): Ability to function effectively on a team whose members together provide leadership, create a collaborative and inclusive environment, establish goals, plan tasks, and meet objectives, and SO (7): Ability to acquire and apply new knowledge as needed, using appropriate learning strategies. Student reporting requirements and the necessary rubrics are incorporated to enable the assessment of the various performance indices for the ABET SO's.

Introduction

Traditionally, undergraduate computational fluid dynamics and heat transfer (CFDHT) courses are designed more as "numerical fluid dynamics and heat transfer" courses, in which the focus is placed on teaching the students the fundamentals of one or more numerical methods, such as the finite difference method (FDM), the finite element method (FEM), and the finite volume method (FVM), and how to use them to solve the differential equations that govern fluid flow and heat transfer. The objectives of many CFDHT courses are to enable the students to write a piece of their own code and use it to simulate benchmark fluid mechanics and heat transfer problems characterized by simplified physics, idealized flow conditions, and small computational domain sizes. This greatly limits the ability of the students to apply what they learn to solve problems of practical industrial applications.

Efforts have been made to close this gap. Recently, with the development of open-source packages such as OpenFoam [1] and the availability of free or low-cost academic licenses for commercial software packages such as ANSYS Fluent [2], some CFDHT courses are now asking the students to simulate more realistic fluid and heat transfer problems. However, due to the limited computational resources available in the classroom, the simulation projects are still limited to simple tutorials and canonical problems packaged with software simulation tools. Moreover, there is usually a steep learning curve for these tools. Only a little effort has been put into teaching the students to understand the effects of the different parameters of the problems, the solvers, and the visualization and post-processing settings required for CFDHT simulations. Often, students can only complete a project by following a tutorial. When an error occurs during the simulation, or the results are inaccurate, students usually do not know how to resolve the issues.

Today's CFDHT engineers are geared more toward using commercial software to solve fluid flow and heat transfer problems with complicated physics and geometries. The computational load is often high and causes a long computational time. This can be remedied by carrying out the computation on high-performance clusters (HPC). However, a high technical barrier exists in equipping the classroom with high-performance computing capabilities. This can also limit the application of complicated but practical fluid flow and heat transfer problems in the CFDHT course.

In our work, we propose a change in the curriculum that addresses the foregoing issues via an introductory CFDHT course that facilitates cloud computing. In addition to introducing the classical numerical methods used in the simulations, the basic theories of high-performance computing will be introduced. Students will be asked to carry out several simulation projects for problems that are relevant to the industry. The commercial package, AEROFLO Cloud, deployed on the cloud and optimized for high-performance computing, will be used as the vehicle for the exercise. Students will use the software package to set up projects, run the simulation on the servers located in the cloud, and post-process the simulation results also on the cloud. This will be done with a classroom PC with an Internet connection. Practical CFDHT simulation details, which are usually not included in the traditional CFDHT course, are briefly presented in the simulation projects. These include mesh quality, high-order spatial and temporal numerical schemes, stability and accuracy of the numerical schemes, turbulence modeling, parallel efficiency, etc.

The learning outcomes of a CFDHT course usually consist of the following:

- To understand the basic numerical methods used in CFDHT calculations.
- To be able to develop a piece of simple CFDHT code.
- To be able to solve simple benchmark fluid dynamics and heat transfer problems using CFDHT.

The new course will also support the following learning outcomes:

- To understand the basic principles of high-performance computing.
- To be able to use commercial software to investigate fluid dynamics and heat transfer problems with complex physics and geometry.
- To be aware of common modeling issues arising during the CFDHT simulations of practical industrial applications.

In the following sections, the contents of the newly designed course will be presented in detail. First, we will provide an introduction to the basics of CFDHT and an overview of the numerical methods utilized in it. Following this, we will provide a comprehensive explanation of the finite difference method (FDM) and offer coding assignments to allow students to practice their skills. Additionally, we will introduce the cloud computing tool used in the course and elaborate on the development of simulation projects that enable students to perform CFDHT calculations for complex fluid problems. Lastly, we will evaluate the new learning outcomes achieved.

Contents of Course

A) Lectures

The new course will be offered at the senior level undergraduate. It will require the students to have taken the basic undergraduate fluid mechanics and heat transfer courses and know a programming language (FORTRAN, MATLAB, C/C++, Python, etc.).

Traditional CFDHT courses place a lot of emphasis on the three numerical methods used in CFDHT: Finite Difference Method (FDM), Finite Volume Method (FVM), and Finite Element Method (FEM). We only focus on the FDM in the newly designed course since the method is simpler. As the first CFDHT course for students, it is more important to understand the basic theory behind it and know how to perform calculations using existing CFDHT tools.

We start the course with an overview of the CFDHT. The general CFDHT procedures to solve engineering problems will be discussed using practical examples, such as calculating the lift and drag force of a NACA0012 airfoil, as shown in Figure 1. Also, CFDHT applications for various engineering fields will be discussed. By doing this, students can be motivated by knowing the possible applications of CFDHT and understanding how a typical CFDHT calculation is done. The limitations of CFDHT are also discussed.

After the introduction, we explain the foundational physics of the problems solved by the CFDHT procedure. The conservation equations governing the fluid dynamics and heat transfer and various types of boundary/initial conditions that can be used to solve the differential equations are reviewed. Standard numerical methods (FDM, FEM, FVM) are introduced. With these discussions, students can understand the precise procedures involved in CFDHT.

We will then focus on the fundamentals of FDM. Students can learn to use finite difference approximation to convert a differential equation into a difference equation, which is then programmed and converted to computer machine language. The various spatial differencing methods (forward, backward, central) and temporal differencing methods (explicit, implicit) are discussed. The way to evaluate numerical errors is also discussed.

We will present the algorithms for solving the system of equations resulting from the differencing procedure. For this purpose, an iterative approach, such as the Jacobi, Gauss-Seidel, and relaxation methods, will be introduced. A sample piece of computer code is given to the student so that they can understand the numerical algorithm. The students are given the first computing assignment (Figure 2) to modify and run the code to solve a one-dimensional steady-state heat conduction problem. Students also learn how to apply various boundary conditions to the algorithm. In the assignment, students are asked to evaluate the numerical errors by comparing the FDM results with the exact solution. Students will also be asked to experiment with different mesh sizes to see how the numerical errors are affected.

The FDM application will be extended to solve transient differential equations. The difference between the explicit (using forward differencing) and implicit (using backward differencing) time marching schemes are discussed. The algorithm of the Newton-Raphson method is discussed to enable the students to use it to solve a set of implicit differencing equations. The second computing assignment (Figure 3) is then given to the students to solve a one-dimensional transient heat conduction problem using both explicit and implicit methods.

Next, we will discuss the implementation of FDM in both two-dimensional and three-dimensional scenarios. Initially, we will provide the third computing assignment (Figure 4), which requires students to solve a steady-state two-dimensional potential flow problem. Then, we will delve into the procedures for applying FDM to transient multi-dimensional problems. In addition to the standard explicit and implicit procedures, we will introduce the alternating direction implicit (ADI) method. To demonstrate this, we will present the fourth computing assignment (Figure 5), which challenges students to solve a two-dimensional transient heat conduction problem using explicit, implicit, and ADI methods.

After the students can solve a single differential equation using FDM, the discussion is extended to solving the multiple coupled fluid governing equations simultaneously. The FDM will be applied to the full Euler equations. The issue of the pressure-velocity coupling is discussed, and the SIMPLE algorithm is introduced as a solution.

An introduction to high-performance computing is subsequently presented to enable students to understand the need for high-performance computing. Different hardware architectures for parallel computing (such as shared memory, distributed memory, etc.) are discussed, and the corresponding parallel programming languages (such as MPI [4], PMV [5], and OpenMP [6]) are introduced.

The purpose of the class is not to teach the students how to write parallel computing code but to enable them to perform parallel computing using existing CFDHT tools. We use the "domain decomposition" method to explain how a calculation can be carried out using multiple processes. We use the Message Passing Interface (MPI) protocol to explain the coupling between the neighboring blocks running on different processes (Figure 6). We teach the students how to evaluate the speed-up, efficiency, and scalability of parallel computing and strategy to achieve good load balancing.

At this point, we are at around 2/3 of the semester (week 10). The remaining part of the course concerns simulation projects. Students will use AEROFLO Cloud [3], which is running on the cloud, to perform several CFDHT simulations to solve some practical fluid problems like those that can be found in industrial applications. We also use these projects to discuss the following advanced CFDHT topics: mesh quality, numerical stability, high-order spatial and temporal schemes, turbulence modeling, and parallel efficiency.

B) Simulation Projects

AEROFLO Cloud, developed by TTC Technologies, Inc., is one of the first commercial CFDHT packages facilitating cloud computing technology. The tool is deployed on a cloud server and optimized for high-performance computing. It is operated based on a Software-as-a-Service model (SaaS) and can be accessed anywhere through a web browser. The interface and the capabilities of the tool are shown in Figure 7.

Three simulation projects are designed. The first project allows students to learn how to use AEROFLO Cloud to perform an aerodynamic airflow simulation inside a convergent-divergent nozzle. The description of the project can be found in Figure 8. Through the project, students can get familiar with the process of a CFDHT application: pre-processing, simulation, and post-processing. The concepts for the computational mesh quality (minimum mesh size, skewness, adjacent mesh size ratio, etc.) will be introduced during the projects, and students are asked to run the same simulations with computational mesh grids of different qualities to evaluate how the mesh quality can affect the accuracy of the results. Students are also asked to run the simulation using different time step sizes to see how its value can affect the convergence speed as well as the numerical stability of the CFDHT solver. The necessity of turbulence modeling is explained. Students are asked to try different turbulence models, such as one-equation, Spalart-Allmaras, two-equation k-ϵ, and two-equation k-ω models for the simulation.

The second project will teach students how to run a CFDHT calculation using the parallel computing technique. In this project, the study of the aerodynamics of an Onera M6 airfoil can be run with three parallel processes using the “domain decomposition” method, as shown in Figure 9. The different block coupling types, such as direct and overset coupling, are discussed using the given mesh grids. A single-block version of the mesh grid is also available so that students can use it as a baseline to evaluate the speed-up of the 3-block parallel calculation. The concepts of high-order and upwind spatial schemes are introduced, and students are asked to run the calculation with different spatial schemes (such as the upwind Roe, MUSCL, and WENO schemes) to compare the results. Similarly, the high-order temporal scheme (such as the Runge-Kutta and Beam-Warming schemes) is also introduced to the students so they can try it in AEROFLO Cloud.

The last project asks the student to use all they have learned to set up and run a simulation of a large and complex problem – simulating the aerodynamics of a commercial Boeing-747 aircraft. Students are permitted to form groups consisting of two members to work together. Due to the complex geometry of the problem, the computational grid provided to students consists of nine blocks, as shown in Figure 10. Students must determine the correct boundary conditions for each block and use appropriate simulation parameters. Usually, students can meet many issues and need to go through multiple try-and-error cycles. Individual instructions are given to explain why the error happens and how it should be fixed. Ultimately, students should gain experience running a practical, large-scale CFDHT simulation using commercial software packages.

Assessment of Learning Outcomes

The proposed course will contribute to three ABET student outcomes (SOs): SO (1): An ability to identify, formulate, and solve complex engineering problems by applying principles of engineering, science, and mathematics, SO (5): An ability to function effectively on a team whose members together provide leadership, create a collaborative and inclusive environment, establish goals, plan tasks, and meet objectives, and SO (7): An ability to acquire and apply new knowledge as needed, using appropriate learning strategies.

For each SO, we will design several performance indicators (PI), each of which will be assessed via a five-scale rubric. The proposed PIs for SO (1) are as follows:

- 1a) Select an appropriate model for the problem.
- 1b) Prepare a solution that exhibits a logical sequence of steps consistent with the model.
- 1c) Demonstrate a correct solution to the problem.
- 1d) Present the solution in the appropriate format.

For SO (5), we are proposing the PI's:

- 5a) Develops team goals, define criteria for conflict resolution, and communicate team decisions.
- 5b) Completes tasks on time.
- 5c) Monitors group progress.
- 5d) Initiates interactions with other team members and facilitates interaction between team members.
- 5e) Evaluates the work of peers,

while for SO (7), we are considering the PI's:

- 7a) Take the initiative to find appropriate resources related to the new knowledge needed based on the objectives and the problems that may encounter during a project or course.
- 7b) Learn new knowledge based on the current knowledge using appropriate learning strategies, such as independent study, brainstorming, discussion, or consultation.
- 7c) Apply the new knowledge to solve the problems and fulfill the objectives during the course of the project.

For each PI of an SO, we will have a five-item rubric. The five items in the rubrics are: 5=Exemplary, 4=Good, 3=Adequate, 2=Marginal, and 1=Unacceptable. Based on student and/or group material submissions, we will evaluate the extent to which a PI is met on a scale of 1 through 5. We will then average the scores for each PI over all students or teams. These averages, together with the number of students that scored above a program-agreed threshold for acceptable performance, will be submitted to the program ABET coordinator for a particular SO, who will synchronize the PI data from the CFDHT course with the PI data from the other courses in the program that contribute to the particular SO. The program will then be able to assess whether an ABET outcome is met for an SO.

As an example, PI (1a: Appropriate Model) of SO (1) could have the following rubrics: 5= Best model is selected for the problem, 4=A correct model is selected, 3=A correct model is chosen, but there are some conceptual errors, 2= Incorrect model is selected for the problem, and 1= No model is selected for the problem. The materials collected from the students will be used to grade the rubrics and assign scores.

Also, PI (1a: Team Goals) of SO (5) may have the rubrics: 5 = Individual contributes a significant amount to the formation of team goals and assumes team leadership, 4=Individual contributes a significant amount to the formation of team goals, 3=Individual contributes an appropriate amount to the formation of team goals, 2=Individual contributes minimally to the formation of team goals, and 1= Individual does not contribute to the formation of team goals. This latter PI could conveniently be assessed by an independent survey of the team members.

Conclusion

A new course in the undergraduate curriculum that teaches practical aspects of computational fluid dynamics and heat transfer (CFDHT) is proposed in this paper. The course is designed to avoid the shortcomings of CFDHT courses that have been proposed in the past. Instruction delivery based on cloud-based high-performance computing CFDHT tools that take advantage of massive parallelization is at the heart of the proposed course. Simulation projects of different levels of difficulty will be assigned, which will culminate in the analysis of large-scale realistic problems of industrial applications. The proposed course will contribute to ABET's Student Outcomes 1, 5, and 7 via the development of the necessary performance indicators that will be assessed in the course.

As computing technology continues to advance and increasingly sophisticated commercial CFDHT software packages become more popular in engineering and technology applications, we are incorporating relevant instruction into our standard undergraduate CFDHT course to equip students with the necessary skills to apply CFDHT software to real-world problems once they graduate and take on industry employment. Furthermore, the course can serve as a valuable foundation for students pursuing CFDHT in greater depth at the graduate level, providing a solid starting point for their continued study.

References

[1] OpenFoam. https://www.openfoam.com/
[2] ANSYS Fluent. https://www.ansys.com/products/fluids/ansys-fluent
[3] AEROFLO Cloud. http://www.ttctech.com/Aeroflo.aspx
[4] MPI. https://www.mcs.anl.gov/research/projects/mpi/
[5] PVM. https://www.csm.ornl.gov/pvm/
[6] OpenMP. https://www.openmp.org/

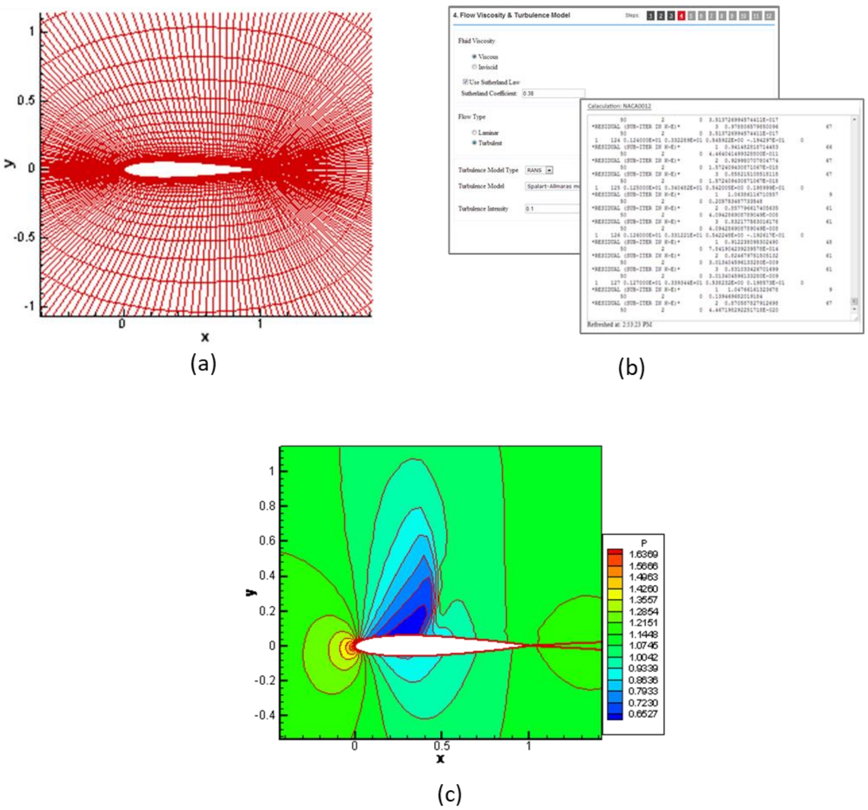
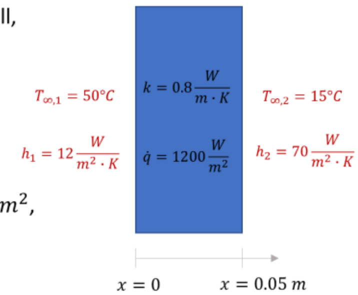


Fig. 1. Using the CFDHT simulation of an airfoil to illustrate the general procedures for a CFDHT calculation: (a) pre-processing, (b) CFDHT simulation, and (c) post-processing.

Computing Assignment #1: **One-Dimensional Steady-State Heat Conduction With Heat Sources**

For a plane wall with internal heat that is generated uniformly throughout this wall, the wall temperature distribution along the x-direction can be solved by

$$\frac{d^2T}{dx^2} + \frac{\dot{q}}{k} = 0$$



Assume
- the thickness of the wall is 5 cm,
- the rate of heat generation per unit surface area of the wall is $\dot{q} = 1200\ W/m^2$,
- the thermal conductivity of the wall material is $k = 0.8\ W/(m \cdot K)$,
- for the air on the left, $T_{f,1} = 20°C$, and $h_1 = 12W/(m^2 \cdot K)$,
- for the air on the right, $T_{f,2} = -15°C$, and $h_2 = 70W/(m^2 \cdot K)$,

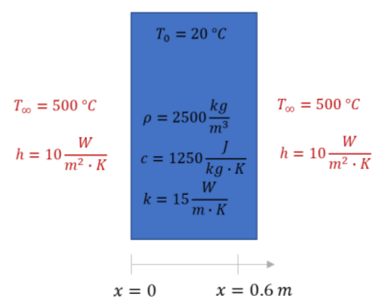
Please use the FDM to solve for $T(x)$ using the following procedures

(1) (20 Pts) Derive the difference equation.
(2) (10 Pts) Apply the correct boundary conditions to the difference equation.
(3) (30 Pts) Program an FDM code to solve using the Gauss-Seidel algorithm.
(4) (20 Pts) Run the code with 5, 10, and 20 equally spaced mesh nodes.
(5) (10 Pts) Derive the analytical solution and evaluate the numerical errors by comparing it with the FDM solution.
(6) (10 Pts) Describe the new theories and methods acquired from the lecture and applied or validated in this project.

Fig. 2. The FDM computing assignment #1: 1D steady-state heat conduction with heat sources.

Computing Assignment #2: **One-Dimensional Transient Heat Conduction**

A 60-cm-thick plane wall at a uniform temperature of 20 $°C$ is suddenly exposed on both sides to a hot gas steam at 500 $°C$. The heat transfer coefficient $h = 10\ W/(m^2 \cdot K)$. The density, specific heat, and thermal conductivity of the wall are $\rho = 2500\ kg/m^3$, $c_p = 1250\ J/(kg \cdot K)$, and $k = 15\ W/(m \cdot K)$. Use FDM, find the temperature distribution inside the plane after 24 hour of heating.



Note that the governing equation for 1D transient heat conduction is

$$\frac{\partial T}{\partial t} = \alpha \frac{d^2T}{dx^2} \qquad \text{where} \qquad \alpha = \frac{k}{\rho c_p}$$

Please use the FDM to solve for $T(x, t)$ using the following procedures

(1) Derive the difference equation using
  a) (15 Pts) the explicit method, and
  b) (15 Pts) the implicit method.
(2) (10 Pts) Apply the correct boundary conditions to the difference equation.
(3) Program an FDM code to solve using both
  (1) (20 Pts) the explicit method, and
  (2) (20 Pts) the implicit method
(4) (10 Pts) Compare the results obtained above.
(5) (10 Pts) Describe the new theories and methods acquired from the lecture and applied or validated in this project.

Fig. 3. The FDM computing assignment #2: 1D transient heat conduction.

Computing Assignment #3: Two-Dimensional Steady-State Potential Flow

Introduction to 2D Potential Flow

For two-dimensional potential (incompressible, irrotational) flow, the flow governing equations reduce to

$$\nabla \cdot \vec{v} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \ \ \text{(continuity)}$$

$$\nabla \times \vec{v} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \ \ \text{(voticity)}$$

where $\vec{v}$ is the velocity vector and (u, v) are its x and y components. By introducing the concept of stream function ϕ, which is defined as

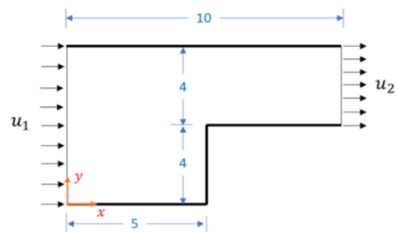
$$u = \frac{\partial \phi}{\partial y}, \qquad v = -\frac{\partial \phi}{\partial x}$$

The continuity equation can be automatically satisfied, and the problem is governed by a single PDE:

$$\nabla^2 \phi = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0 \ \ \text{(Laplace)}$$

Problem Description

Consider a 2D potential flow into a step contraction, as shown in the figure on the right. u_1 and u_2 are the uniform inlet and outlet flow velocities. Knowing $u_1 = 7$, use FDM to solve the fluid dynamics. Please use equally spaced Cartesian mesh nodes and $\Delta x = \Delta y = 1$.



Hints

- u_2 needs to be calculated first through mass conservation.
- Assume $\phi = 0$ at the bottom wall, and $\phi = 49$ at the top wall.

Submission

Compose a report including the following:

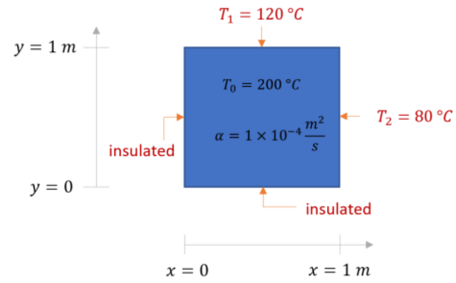
1. (20 Pts) Compose finite difference equations for all mesh nodes.
2. (10 Pts) Find and apply all boundary conditions.
3. (20 Pts) Solve the finite difference equations and find ϕ on each node point.
4. (20 Pts) Plot the stream function contour (i.e., the streamlines).
5. (20 Pts) Compute and find the (u, v) velocities at the locations defined in the table on the right.
6. (10 Pts) Describe the new theories and methods acquired from the lecture and applied or validated in this project.

	x	y	u	v
1	0.0	4.0		
2	2.0	2.0		
3	2.0	6.0		
4	4.0	2.0		
5	4.0	6.0		
6	5.0	6.0		
7	7.5	6.0		
8	10.0	6.0		

Fig. 4. The FDM computing assignment #3: 2D steady-state potential flow.

Computing Assignment #4: **Two-Dimensional Transient Heat Conduction**

The initial temperature of the square bar ($1\text{m} \times 1\text{m}$) is $200°C$, and then suddenly placed into an environment as shown in the figure, where the left and bottom sides are fully insulated, the top and right sides are maintained with a constant temperature of $T_1 = 120\ °C$ and $T_2 = 80\ °C$, respectively. The thermal diffusivity of the plate is $\alpha = 1 \times 10^{-4}\ m^2/s$. Use FDM, find the temperature of the bar at $t = 100\ s$.



Note that the governing equation for 2D transient heat conduction is

$$\frac{1}{\alpha}\frac{\partial T}{\partial t} = \frac{d^2T}{dx^2} + \frac{d^2T}{dy^2}$$

Please use the FDM to solve for $T(x, t)$ using the following procedures

(1) Derive the difference equation using
  a) (10 Pts) the explicit method,
  b) (10 Pts) fully implicit method, and
  c) (10 Pts) ADI methods.
(2) (10 Pts) Apply the correct boundary conditions to the difference equation.
(3) Program an FDM code to solve using
  a) (10 Pts) the explicit method,
  b) (15 Pts) fully implicit method, and
  c) (15 Pts) ADI methods.
(4) (10 Pts) Compare the results obtained above.
(5) (10 Pts) Describe the new theories and methods acquired from the lecture and applied or validated in this project.

Fig. 5. The FDM computing assignment #4: 2D transient heat conduction.

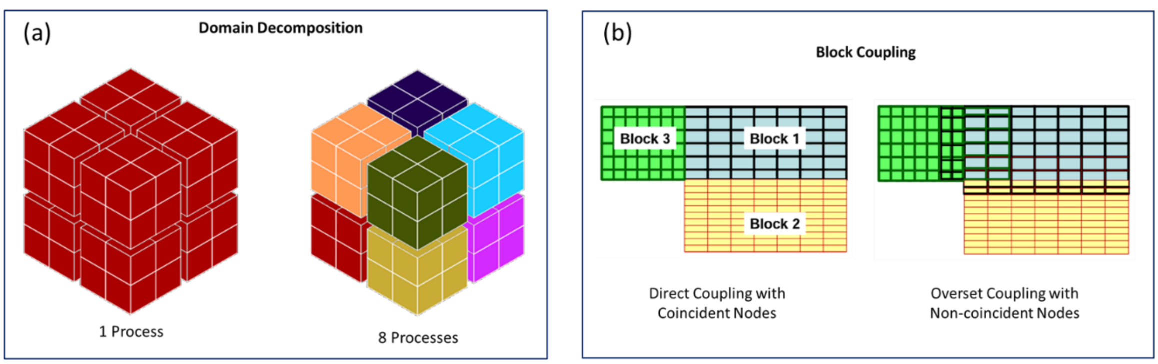
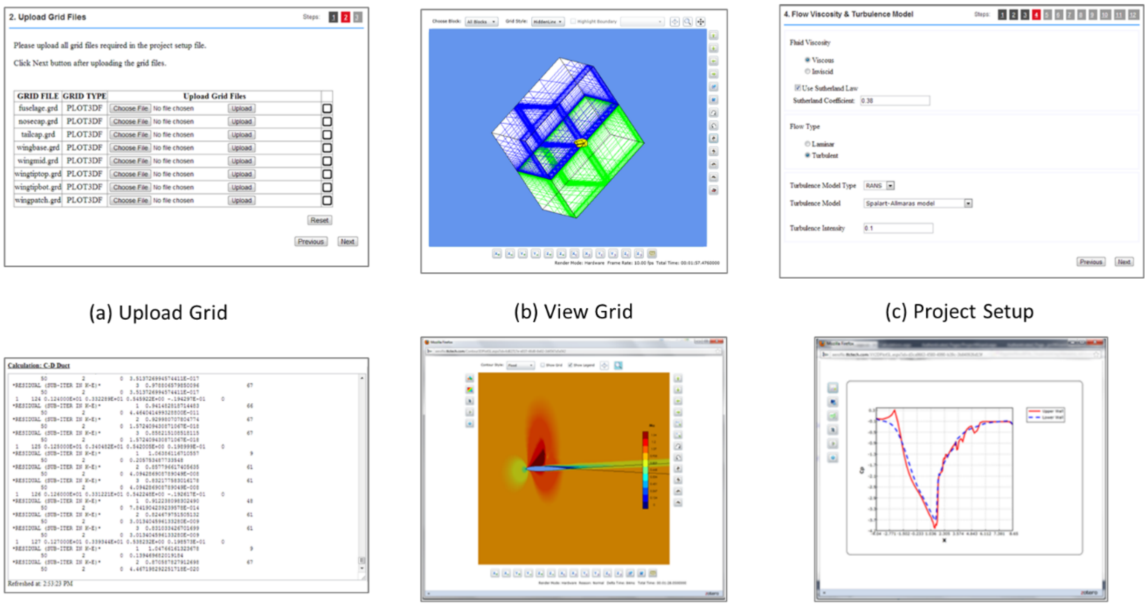


Fig. 6. Parallel CFDHT computing: (a) the concept of domain decomposition, (b) the different coupling methods between the blocks.



(a) Upload Grid (b) View Grid (c) Project Setup

(d) Submit and Monitor the Calculation (e) View Simulation Results (f) Postprocessing Analysis

Fig. 7. Features and user interface of AEROFLO Cloud

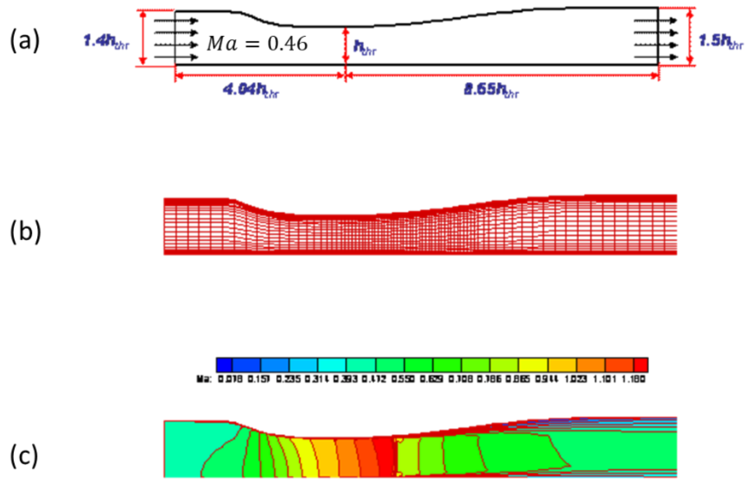


Fig. 8. Simulation project #1: airflow in a convergent-divergent nozzle: (a) nozzle geometry and flow conditions, (b) computational mesh, and (c) calculation result – Mach number contours

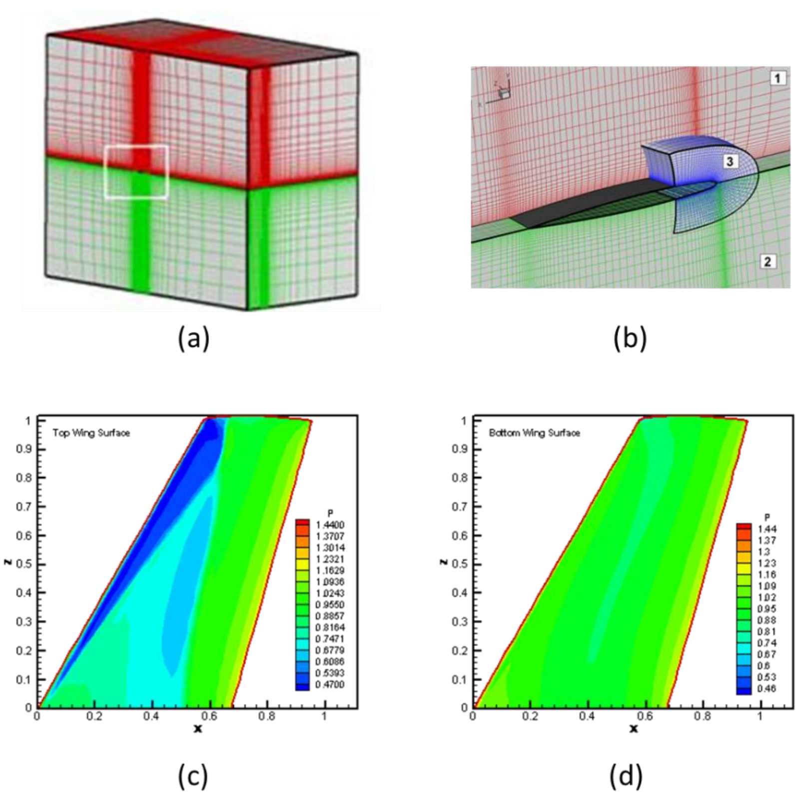


Fig. 9. Simulation project #2: aerodynamics of Onera M6 wing (Mach number = 0.84, attack angle = 3°): (a) computational mesh with three blocks, (b) close view of the computational mesh around the foil, (c) calculation result – pressure contours on the top side of foil surface, and (d) calculation result – pressure contours on the bottom side of foil surface.

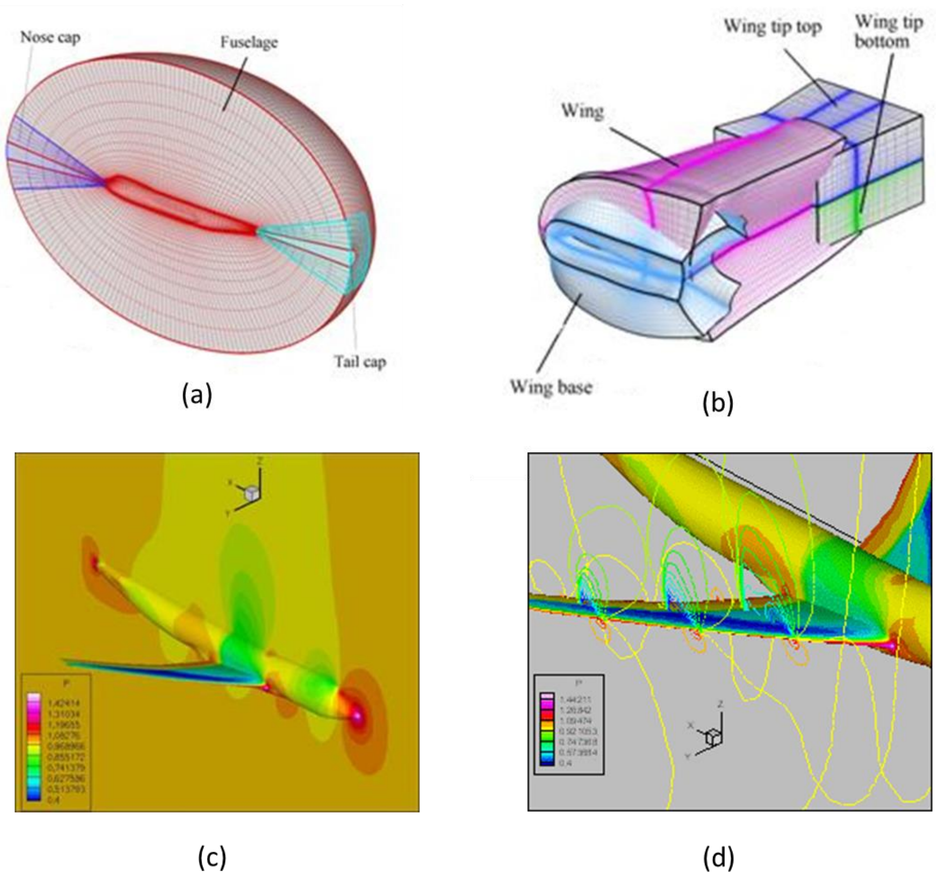


Fig. 10. Simulation project #3: aerodynamics of Boeing 747 (Mach number = 0.85, attack angle = 3°): (a) computational mesh with nine blocks, (b) close view of the computational mesh around the wing, (c) calculation result – pressure contours, and (d) calculation result – pressure line plots around the wing.

Date	**Topic**	**Student Learning Outcomes**
Week 1	Overview of CFDHT	Students will be able to get familiar with the basic concepts, applications, and procedures of CFDHT.
Week 2	Governing Equations	Students will be familiar with the mathematical equations that govern fluid dynamics and heat transfer.
Week 3	FDM: 1D Steady-State Problem	Students will be able to derive finite difference equations for one-dimensional steady-state differential equations.
Week 4	Solutions to a System of Linear Equations	Students will be able to write a program that uses the Jacobi and Gauss-Seidel methods to solve a system of linear equations.
Week 5	FDM: 1D Transient Problem	Students will be able to apply FDM to solve one-dimensional transient differential equations using both explicit and implicit procedures.
Week 6	FDM: 2D & 3D Implementations	Students will be able to utilize FDM to solve two- or three-dimensional steady-state or transient differential equations, employing explicit, implicit, and ADI methods.
Week 7	FDM: Euler Equations	Students will be able to apply FDM to investigate incompressible, inviscid flows.
Week 8	Pressure-Velocity Coupling	Students will be able to utilize the SIMPLE algorithm to address pressure-velocity coupling in CFDHT.
Week 9	High-Performance Computing	Students will gain an understanding of the fundamental concepts and techniques used in parallel computing.
Week 10	Simulation Project 01: Convergent-Divergent Nozzle	Students will be able to use commercial CFDHT software to model a basic fluid flow problem.
Week 11	Simulation Project 02: Onera M6 Wing	Students will be able to use parallel computing to perform simulations using commercial CFDHT software.
Week 12	Simulation Project 03: Boeing 747	Students will be able to apply commercial CFDHT software to address real-world problems with practical applications.

Fig. 11. Proposed course outline.

Attendance	5%
Computing Assignment 1	10%
Computing Assignment 2	10%
Computing Assignment 3	10%
Computing Assignment 4	15%
Simulation Project 1	15%
Simulation Project 2	15%
Simulation Project 3	20%
Total	**100%**

Fig. 12. Grading scheme.

Simulation Project 01 - Convergent-Divergent Nozzle

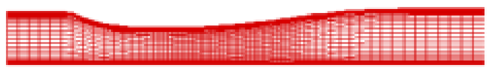
Problem Description

The physical domain and its dimensions are shown in the figure below,

where h_{thr} = 0.14435 ft. This provides a Mach number of 0.46 at the inlet and a Reynolds number of 687,810.4. The total inlet pressure and outlet pressures are 19.58 psi and 16.05 psi, respectively, and the total temperature is 500 °R. Uniform velocity should be applied at the inlet. The no-slip wall boundary conditions are imposed at the top and bottom walls. Dirichlet pressure boundary conditions are applied at the inlet and outlet, while zero Neumann boundary conditions are applied at the outlet for the other flow variables.

Computational Grid

Two grids of different mesh quality (cdvduct1.PLOT3D, cdvduct2.PLOT3D) can be used to calculate flow through the duct with grid clustering in the vicinity of solid walls. The mesh geometry is shown in the figure below.



CFD Simulation

Suggested simulation parameters are given below.

Turbulence Model	Spalart-Allmaras
Spatial Scheme	MUSCL
Temporal Scheme	Beam-Warming
Time Step Size	0.001
Number of Steps	50000

More details can be found at http://www.ttctech.com/Samples/CDNozzle/cdvduct.htm

Complete the following tasks:

- Set up using the first mesh grid and simulate it with the suggested simulation parameters above. Monitor the residual plot to check the convergence of the simulation.
- Analyze the simulation results. Generate contour plots of the Mach number and profiles of the pressure line on the top and bottom walls.
- Repeat the simulation with the second mesh grid. Compare the results of different mesh grids.
- Repeat the simulation by switching the turbulence models to (a) $k-\epsilon$ model and (b) $k-\omega$ model. Compare the results of different turbulence models.
- Repeat the simulation by increasing the time step size to 0.005, 0.01, and 0.05 and decreasing the time steps to 10000, 5000, and 1000, respectively. Report the simulation results or any errors encountered.

Report Submission

Compose a lab report including the following:

1. (10 Pts) Describe the problem to simulate, including the computational domain's geometry and the flow conditions.
2. (10 Pts) Provide details about the two computational grids, including the grid size and the grid quality (These can be obtained from AEROFLO Cloud)
3. (10 Pts) Provide a summary of the simulation parameters used and list the initial and boundary conditions applied.
4. (15 Pts) Provide screenshots depicting each simulation setup step in AEROFLO Cloud.
5. (10 Pts) Create Mach number contours and compare the results of different mesh grids.
6. (10 Pts) Generate pressure profiles for the top and bottom surfaces and compare the results of different mesh grids with experimental data (Exp.xlsx).
7. (10 Pts) Repeat (5) to compare the results of various turbulence models.
8. (10 Pts) Repeat (6) to compare the results of different turbulence models.
9. (15 Pts) Please briefly summarize the conclusions drawn from the obtained results. Additionally, describe the key takeaways from the project, the challenges encountered, and the strategies employed to overcome them.

Fig. 13. Simulation project #1 handout

Simulation Project 02 - ONERA M6 Wing

Problem Description

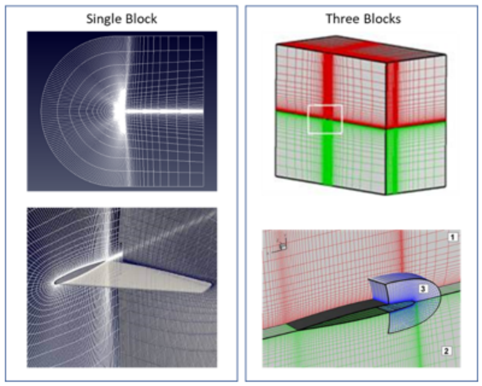
The transonic flow around the ONERA-M6-Wing for a Mach number of 0.84 and an angle of attack of 3° needs to be simulated. The shape of the wing is shown below. The non-slip wall conditions are set on all foil surfaces.

Computational Grid

Two sets of computational grids are available:

1. The single-block grid (M6_1B.Plot3D) with a grid size of 189×40×50.
2. The three-block grid (M6_3B.Plot3D) with grids size of 99×57×33, 99×57×33, and 61×49×33

The mesh geometry is shown in the figure below



CFD Simulation

Suggested simulation parameters for the simulations are given below.

Viscosity	Inviscid
Spatial Scheme	MUSCL
Temporal Scheme	Beam-Warming
Time Step Size	0.001
Number of Steps	25000

More details can be found at http://www.ttctech.com/Samples/Onera_M6_Wing/onera_sb.htm and http://www.ttctech.com/Samples/Onera_M6_Wing_Multiblock/onera_mb.htm

Complete the following tasks:

- Set up and run the 1-block and 3-block simulations with the suggested simulation parameters.
- Repeat the 3-block simulation by switching the spatial scheme to (a) upwind Roe and (b) WENO schemes. Compare results of different spatial schemes.
- Repeat the 3-block simulation by switching the temporal scheme to the Rugne-Kutta scheme. Compare results of different temporal schemes.

Report Submission

Compose a lab report including the following:

1. (10 Pts) Describe the problem to simulate, including the computational domain's geometry and the flow conditions.
2. (15 Pts) Provide a summary of the simulation parameters used and list the initial and boundary conditions applied for the 3-block simulation.
3. (15 Pts) Provide screenshots depicting each simulation setup step in AEROFLO Cloud for both the 3-block simulation.
4. (15 Pts) Produce pressure contours on the foil's top and bottom surface and compare the results obtained from the 1-block and 3-block simulations.
5. (15 Pts) Estimate the speed-up and parallel efficiency of the 3-block simulation.
6. (15 Pts) Generate pressure coefficient profiles for the top and bottom surfaces at the 65% span location and compare the results obtained from different temporal and spatial schemes with experimental data (Cp65.xlsx).
7. (15 Pts) Please briefly summarize the conclusions drawn from the obtained results. Additionally, describe the key takeaways from the project, the challenges encountered, and the strategies employed to overcome them.

Fig. 14. Simulation project #2 handout

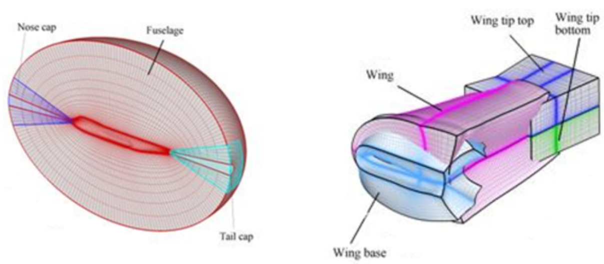
Simulation Project 03 – Boeing 747

Problem Description

The flow around the Boeing 747-200 aircraft (figure below) needs to be simulated at cruise conditions using a multi-block grid. The following conditions were used: $M_\infty = 0.855$, a = 3.05°, reference area = 5500 sq ft (792,000 sq in), moment center = (1339.91, 0., 191.87) in., moment reference length = 327.8 in. The spatial dimensions have been normalized with the moment reference length, leading to a reference Reynolds number, Re = 3.5×10^6.

Computational Grid

The computational grids (b747.Plot3D) contained nine blocks with the following grid points: fuselage 138×70×30=416,000, nose cone 31×20×30=18,600, tail cap (31×20×30=18,600), wing base 129×38×30=147,060, wing mid-section 50×129×29=187,050, wing tip (top) 77×41×28 = 81,508, wing tip (bottom) 77×41×28=81,508, wing patch 71×71×71=357,911, and far-field grid 73×39×48=136,656. This yields a total number of grid points of 1,444,993. The mesh geometry is shown in the figure below.



CFD Simulation

Suggested simulation parameters for the simulations are given below.

Viscosity	Inviscid
Spatial Scheme	MUSCL
Temporal Scheme	Beam-Warming
Time Step Size	1×10^{-4}
Number of Steps	80000

More details can be found at http://www.ttctech.com/Samples/b747/b747.htm

Complete the following tasks:

- Figure out the connectivity between the blocks and find the correct boundary conditions for each block.
- Set up and run the simulation with the suggested simulation parameters.

Report Submission

Compose a lab report including the following:

1. (10 Pts) Describe the problem to simulate, including the computational domain's geometry and the flow conditions.
2. (15 Pts) Provide details about the computational grid, including the grid size and the block connectivity.
3. (15 Pts) List the boundary conditions for each block.
4. (15 Pts) Provide a summary of the simulation parameters used and list the initial conditions applied.
5. (15 Pts) Provide screenshots depicting each simulation setup step in AEROFLO Cloud.
6. (15 Pts) Plot pressure contour on the airplane surface.
7. (15 Pts) Please briefly summarize the conclusions drawn from the obtained results. Additionally, describe the key takeaways from the project, the challenges encountered, and the strategies employed to overcome them.

Fig. 15. Simulation project #3 handout